11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
E. Oñate, J. Oliver and A. Huerta (Eds)

ON THE CONTROL OF SPURIOUS FORCE OSCILLATIONS FOR MOVING BODY PROBLEMS USING AN IMMERSED BOUNDARY METHOD

M. BELLIARD[◇,*], M. CHANDESRIS[◇,†], J. DUMAS[◇,‡], Y. GORSSE[◇,*], D. JAMET[◇,‡] AND C. JOSSERAND[♮]

[◇] Commissariat à l'Énergie Atomique et aux Énergies Alternatives, CEA, DEN/DANS/DM2S/STMF/LMEC

[*]CEA Cadarache, bâtiment 238, F-13108 Saint Paul-lez-Durance, France, michel.belliard@cea.fr, yannick.gorsse@cea.fr

[†] CEA Grenoble, bâtiment 10.05, F-38054 Grenoble Cedex, France

[‡] CEA Saclay, bâtiment 454, Route Nationale, F-91400 Gif-sur-Yvette Cedex, France

[♮] Institut Jean le Rond d'Alembert, Université, Pierre et Marie Curie, Tour 55-65, 4 place Jussieu, F-75252 Paris Cedex 05, France

Key words: Immersed Boundary Method, Moving Bodies, Spurious Force Oscillations, Fluid Structure Interaction

Abstract. We present a Direct Forcing (DF) approach for simulations of flows over moving bodies, on Cartesian grids, capable of reducing the amplitude of the force oscillations, by using a regularized approach.

1 INTRODUCTION

Fluid-structure interaction (FSI) problems are tricky to simulate due to the complexity of the physical phenomena and the presence of time varying geometries. In the well-known body-fitted approach the boundary conditions (BC), that are critical for FSI issues, are exactly imposed on the physical domain boundary. However accurate this approach may be, it is sometimes difficult to deal with the re-meshing issue, specially for large fluid-mesh deformations. Another approach consists in using immersed boundary methods (IBM) [1]. The general idea is to consider only a fluid domain in which the solid boundaries are immersed. We recover the immersed BC by adding a supplementary term in the governing equations of the fluid, referred to as the forcing term. Here, we consider the Direct Forcing (DF) method [2] that is particular discrete version of the IBM for rigid

solids. This method has been successfully applied to various problems [3]. Unfortunately, one undesirable property is the generation of spurious force oscillations (SFO) [4, 5, 6] when dealing with moving bodies on a fixed computational grid.
Here, we present a Regularized Direct Forcing (RDF) approach [7] for simulations of flows over moving bodies capable of reducing the amplitude of the SFO. We have identified the primary source of these oscillations as the temporal discontinuity of the forcing term across the fluid-solid interface, concerning mainly the dead cells. Indeed, when the velocity point comes near the solid boundary, the original DF approach consists in forcing the previously computed fluid velocity to a prescribed one. Therefore the forcing term is highly discontinuous in time for dead cells leading to the famous spurious force oscillations. In the proposed RDF approach, this prescribed fluid velocity is chosen in a such a way that the magnitude of the forcing term tends to zero with the distance to the immersed interface (or the solid volume fraction). By modifying the definition of the prescribed fluid velocity, it is possible to change the order of approximation of the method. A first order and a second order version of the method are presented here.
The results obtained with the present RDF approach are similar to those obtained with much more complicated method [8, 6, 9]. Moreover this method is very easy to implement, effective and improve the numerical precision by comparison with the original DF approach.

2 NUMERICAL METHODS

The governing equations used to describe unsteady incompressible flows are the incompressible Navier-Stokes (NS) equations -here with Dirichlet BC for the sake of the presentation-:

$$\begin{aligned} \frac{\partial \underline{u}}{\partial t} + \nabla \cdot (\underline{u} \otimes \underline{u}) + \frac{1}{\rho}\underline{\nabla} p - \nu \nabla^2 \underline{u} &= \underline{0} \quad \text{in } \Omega \\ \nabla \cdot \underline{u} &= \underline{g} \quad \text{in } \Omega \\ \underline{u} &= \underline{u}_D \text{ on } \partial\Omega \end{aligned} \tag{1}$$

where Ω is the computational domain, $\partial\Omega$ its boundary, $\underline{u}$ the fluid velocity, ν the kinematic viscosity, p the pressure, ρ the fluid density and $\underline{g}$ bulk forces. A non-incremental fractional-step scheme is used to solve the governing equations (1) [10, 11].

Following the IBM developed by [2], the computational domain Ω corresponds to a uniform Eulerian grid including the fluid domain Ω_F and the embedded solid domain Ω_S such as $\Omega = \Omega_F \cup \Omega_S$. The presence of embedded time-varying geometry Ω_S is taken into account thanks to a source term $\underline{F}_{DF}$, added to the time semi-discretized form of the Navier-Stokes Eqs. (1). For instance, with an explicit Euler scheme and considering

a projection solver, this is done in two steps:

$$\frac{\underline{u}^\star - \underline{u}^n}{\Delta t} + \nabla \cdot (\underline{u}^n \otimes \underline{u}^n) + \underline{\nabla} P^n - \nu \nabla^2 \underline{u}^n \;=\; \underline{0} \text{ in } \Omega, \tag{2}$$

with $\underline{u}^\star$ a predicted velocity without taken into account the immersed boundary Σ. This step depends on the time scheme. The forcing step is:

$$\underline{\tilde{u}} \;=\; \underline{u}^\star + \Delta t \underline{F}_{DF} \tag{3}$$

where $\underline{\tilde{u}}$ is the predicted velocity that satisfied the IBC and Δt the time step. Finally, the projection and the correction steps remains classical:

$$\frac{1}{\rho} \Delta p^{n+1} \;=\; \frac{1}{\Delta t} \nabla \cdot \underline{\tilde{u}} \text{ in } \Omega \tag{4}$$

$$\underline{u}^{n+1} = \underline{\tilde{u}} - \frac{\Delta t}{\rho} \underline{\nabla} p^{n+1} \text{ in } \Omega \tag{5}$$

The source term $\underline{F}_{DF}$ is defined by:

$$\underline{F}_{DF} = \chi_{\mathrm{m}} \frac{\underline{u_i} - \underline{u}^\star}{\Delta t} \text{ in } \Omega \tag{6}$$

where $\underline{u_i}$ is the imposed fluid velocity and χ_{m} is the phase indicator function of the imposed velocities domain Ω_I^{m}:

$$\chi_{\mathrm{m}}(\underline{x}) = \begin{cases} 1 & \text{for } \underline{x} \in \Omega_I^{\mathrm{m}} \\ 0 & \text{otherwise.} \end{cases} \tag{7}$$

where $\underline{x}$ is the position. The imposed fluid velocity can obtained by interpolations of various orders from the BC and fluid velocity. Here we consider interpolations of order 1 (base model: the imposed velocity is equal to the solid one $\underline{u}_D$) and of order 2 (linear model: linear interpolation [12]).

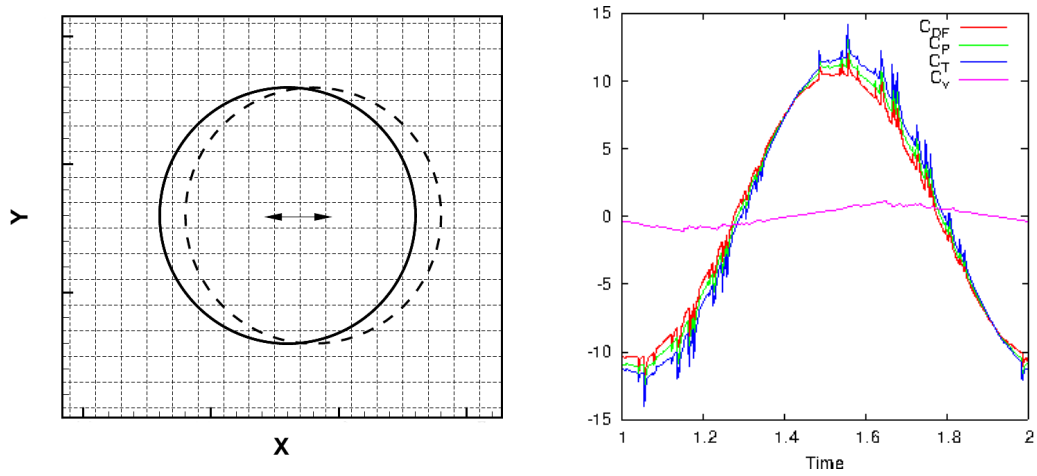
The space discretization is based on a finite volume approximation with a staggered grid arrangement of the variables $(\underline{u}, P)$. As a result, the pressure degrees of freedom are located at the cell centers whereas those of each velocity component are placed at the middle of the cell edges. The governing Eqs. (1) are integrated over each volume control ensuring the conservation of mass and momentum balance. The convection and diffusion terms are respectively approached by the QUICK and centered schemes.

3 ANALYZE OF THE SFO

Let us consider the test case of Seo and Mittal [6] as sandbox. It consists in a circular cylinder of diameter D oscillating along the x-direction in a fluid at rest:

$$\begin{aligned} x_c(t) &= x_c(0) + X_0(1 - cos(2\pi f_0 t)) \\ y_c(t) &= y_c(0) \end{aligned} \tag{8}$$

where $(x_c(0), y_c(0))$ are the initial coordinates of the cylinder center, X_0 the oscillations amplitude and their frequency f_0, see the Fig. 1(a). Fig. 1(b) shows the evolution over a period T_0 of the dimensionless coefficients concerning the pressure drag C_P, the friction drag C_ν, the total drag C_T and the total contribution of the direct forcing term $C_{DF} = \frac{\iiint_{\Omega_S} \rho \underline{F}_{DF} \cdot \underline{e}_x \mathrm{d}\Omega}{\underline{F}_{ref} \cdot \underline{e}_x}$ with $\underline{F}_{ref}$ a scaling vector and $\underline{e_x}$ is the x-direction unit vector.



(a) (b)

Figure 1: Mittal's test case 1(a) geometry and computational grid. Pysical dimensionless coefficients time histories: C_P, C_ν, C_T and C_{DF}.

Mainly it is the pressure drag component that is disturbed by the spurious oscillations whereas the time history of the friction drag is almost regular. Therefore, the SFO are in fact due to the spurious pressure oscillations (SPO). It is also interesting to notice that C_{DF} is nearly equal to C_T.

Analyzing the SPO origin in the case of the DF base model, we obtain under reasonable hypothesis [7] that the measure of the DF term variation between the time steps n and $n+1$ can be approached by:

$$\begin{aligned} \int_{\Omega_{s,h}^{n+1}} \underline{F}_{DF}^{n+1} \mathrm{d}\Omega - \int_{\Omega_{s,h}^{n}} \underline{F}_{DF}^{n} \mathrm{d}\Omega &= \underbrace{\mathcal{O}(1)}_{K \in \Omega_{s,h}^{n+1} \cap \Omega_{s,h}^{n}} + \underbrace{\mathcal{O}(\Delta x) + \mathcal{O}(\frac{\Delta x^2}{\Delta t})}_{K \in DC} + \underbrace{\mathcal{O}(\Delta x)}_{K \in FC} \\ &= \mathcal{O}(1) + \mathcal{O}(\Delta x) + \mathcal{O}(\frac{\Delta x^2}{\Delta t}) \end{aligned} \quad (9)$$

where Δx is the space step, $\Omega_{s,h}^{n*}$ is imposed velocities domain at time $n*$, the FC (fresh cells) is the set of velocities that became freshly fluids ($K \in \Omega_{s,h}^{n} \setminus \Omega_{s,h}^{n+1}$) and DC (dead cells) the set of velocities that became freshly forced ($K \in \Omega_{s,h}^{n+1} \setminus \Omega_{s,h}^{n}$). This equation

implies that the SPO decrease with decreasing the grid spacing and increasing the computational time step. The exponents are in good agreement with [6]. Additional studies conclude that the DC are the main source of pressure oscillations [7].

4 REGULARIZED DIRECT FORCING

The classical DF method consists in modeling χ_m in consistence with a sharp transition of the forcing term at the forced-free interface, cf. Eq. (7). Then when a fluid point is reached by the solid boundary, we shapely jump from a velocity computed by the Navier-Stokes numerical scheme to a geometrical interpolation. Then the forcing term is highly discontinuous in time for the dead cells.
The key idea of the regularized DF formulation is to smooth this transition by replacing χ_m with a smoother function. The following formulation is proposed for the imposed velocity:

$$\underline{u_i} \;=\; \tau_\mathrm{m}\underline{u}_\mathrm{m} + (1-\tau_\mathrm{m})\underline{u}^\star \tag{10}$$

where τ_m is a smooth function of the distance to the immersed boundary Σ. For instance, τ_m can be the fraction of the forced-domain volume to the control volume associated with the fluid velocity.
Hence we force the velocity proportionally to τ_m in order to have a smooth transition: the fluid velocity will be forced progressively to the forced velocity as it enters in the forced domain. This formulation does not increase the computational time cost and is very easy to implement. We named it "regularized" (RG) DF because the imposed velocity is a linear combination of the NS computed velocity and the imposed interpolated velocity leading to a imposed regularized velocity.

For your tests we choose the following model for τ_m:

$$\underline{u_i} = \begin{cases} \underline{u}_\mathrm{m} & \text{if } \tau_\mathrm{m} = 1 \\ \tau_\mathrm{m}\underline{u}_\mathrm{m} + (1-\tau_\mathrm{m})\underline{u}^\star & \text{if } 0 < \tau_\mathrm{m} < 1 \\ \underline{u}^\star & \text{if } \tau_\mathrm{m} = 0 \end{cases} \tag{11}$$

with τ_m computed thanks to the distance function. It varies linearly with the distance from 0 to 1 across a layer of thickness Δx centered on the forced-free interface, see Fig. 2.

Applying similar analysis than done in Section 3, we now obtain that the measure of the DF term variation between the time steps n and $n+1$ can be approached by:

$$\begin{aligned} \int_{\Omega_{s,h}^{n+1}} \underline{F}_{RG}^{n+1}\mathrm{d}\Omega - \int_{\Omega_{s,h}^{n}} \underline{F}_{RG}^{n}\mathrm{d}\Omega \;&=\; \underbrace{\mathcal{O}(1)+\mathcal{O}(\Delta t)+\mathcal{O}(\Delta x)}_{K\in\Omega_{s,h}^{n+1}\cap\Omega_{s,h}^{n}} + \underbrace{\mathcal{O}(\Delta t)+\mathcal{O}(\Delta x)}_{K\in DC} + \underbrace{\mathcal{O}(\Delta t)}_{K\in FC} \\ &=\; \mathcal{O}(1)+\mathcal{O}(\Delta t)+\mathcal{O}(\Delta x) \end{aligned} \tag{12}$$

Hence, the $\mathcal{O}(\frac{\Delta x^2}{\Delta t})$ term of Eq. (9) is reduced to $\mathcal{O}(\Delta x)$.

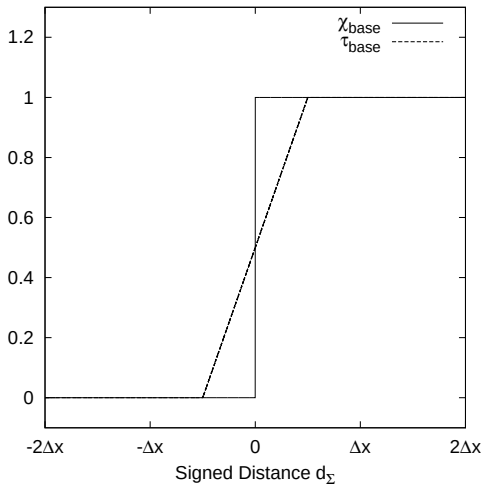


Figure 2: Representation of χ_{m} and τ_{m} for the classical DF method and the Regularized DF one.

5 NUMERICAL APPLICATIONS

To demonstrate the effectiveness of the present DF forcing approach in comparison with the original one, we simulate two classic moving-body problems from the literature by varying both the grid spacing and the computational time step. The first test case is from [6] where we consider an oscillating cylinder in a fluid at rest. Concerning the second test case, it is an oscillating cylinder in a cross flow [13].

The force oscillations are almost entirely suppressed for the first test case, cf Fig. 3.

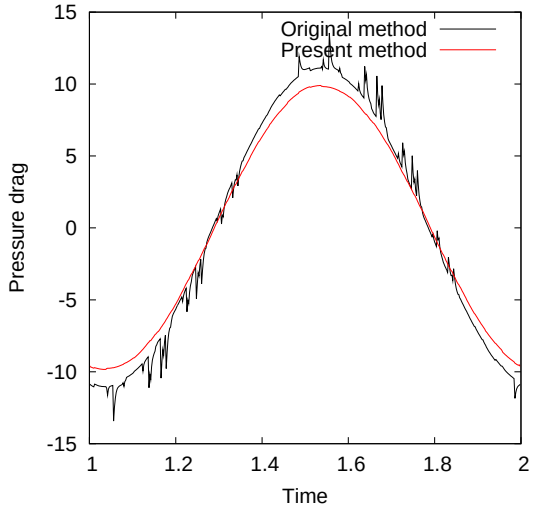
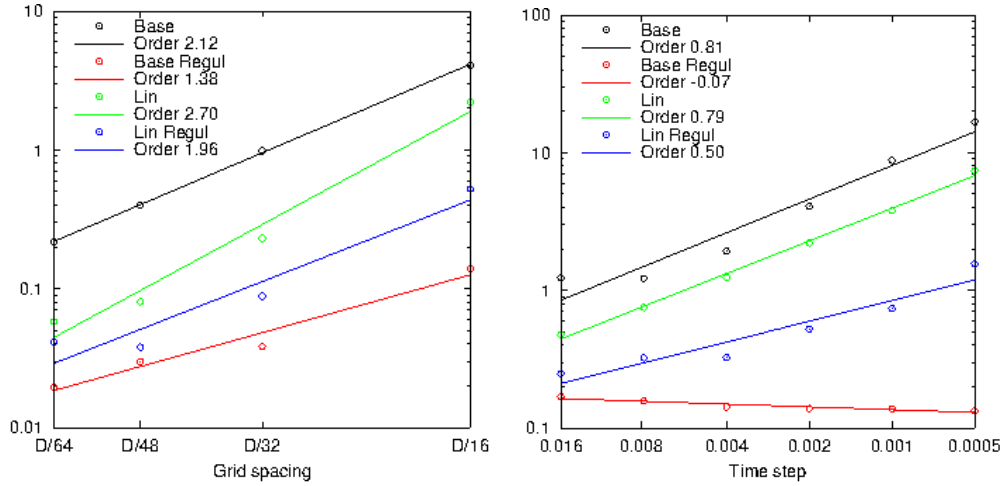


Figure 3: Time evolution of the pressure drag for an oscillating cylinder, test case from [6], using the original method and the present method, both with the first order version.

We used the pressure 2δ-discontinuity $C_P^{2\delta}$ [6] to quantitatively describe the SPO:

$$C_P^{2\delta} = C_P^{n+1} - 2C_P^n + C_P^{n-1} \tag{13}$$

where n is the time-step index. Space-step and time-step convergence studies of the amplitude of the SPO, cf. Fig. 4, conclude to a big damping of the SPO and corroborate the theoretical analysis.



(a) Grid spacing convergence of the oscillations

(b) Time-step convergence of the oscillations

Figure 4: $Cp_{Max}^{2\delta}$ for the base and the linear interpolation schemes for both the DF and RG formulations. The CFl number is set to 1.

Also the present DF approach decreases the amplitude of the oscillations by almost one order of magnitude for the second one.

6 CONCLUSIONS

Consequently, by choosing carefully the grid spacing and the time step it is possible to control and manage the magnitude of the force oscillations, leading possible to simulate full fluid structure interaction problems such as flow induced vibration.

REFERENCES

[1] Peskin C. Numerical Analysis of blood flow in heart. *J. of Comput. Phys.* (1977) **25**:220-252.

[2] Mohd-Yussof J. Combined Immersed Boundaries/B-Splines Methods for Simulations of Flows in Complex Geometries. *CTR Annual Research Briefs, NASA Ames/Stanford University* (1997).

[3] Fadlun E. A., Verzicco R., Orlandi P. and Mohd-Yussof J. Combined immersed-boundary finite-difference for three-dimensional complex flow simulations. *J. of Comput. Phys.* (2000) **161**:35-60.

[4] Yang X., Zhang X., Li Z. and He G. A smoothing technique for discrete delta functions with application to immersed boundary method in moving boundary simulations. *J. of Comput. Phys.* (2009) **228**:7821-7836.

[5] Liao C.-C., Chang Y.-W., Lin C.-A. and McDonough J.M. Simulating flows with moving rigid boundary using immersed-boundary method. *Computers & Fluids* (2010) **39**:152-167.

[6] Seo J.H. and Mittal R. A sharp-interface immersed boundary method with improved mass conservation and reduced spurious pressure oscillations. *J. Comput. Phys.* (2011) **230**:7347–7363.

[7] Belliard M., Chandesris M., Dumas J., Gorsse Y., Jamet D. and Josserand C. A regularized formulation of direct forcing IBM to control spurious force oscillations in order to simulate complex fluid structure interaction problems. *submitted to J. of Comput. Phys.* (2014).

[8] Lee J., Kim J., Choi H. and Yang K.S. Sources of spurious force oscillations from an immersed boundary method for moving-body problems. *J. Comput. Phys.* (2011) **230**:2677–2695.

[9] Lee J. and You D. An implicit ghost-cell immersed boundary method for simultions of moving body problems with control odd spurious force oscillations. *J. Comput. Phys.* (2013) **233**:295–314.

[10] Chorin A.J. Numerical solution of the Navier-Stokes equations. *Math. Comput.* (1968) **22**:745-762.

[11] Temam R. Sur l'approximation de la solution des équations de Navier-Stokes par la méthode des pas fractionnaires. *Arch. Rational Mech.* (1969) **32**:135-153.

[12] Introïni C., Belliard M., Fournier C. A second order penalized direct forcing for hybrid cartesian/immersed boundary flow simulations. *Computers and fluids.* (2014) **90**:21-41.

[13] Guilmineau E. and Queutey P. A numerical simulation of vortex shedding from an oscillating circular cylinder. *J. Fluids and Structures* (2002) **16**:773–794.